AN IMPLICIT DYNAMIC WAVE MODEL FOR MIXED FLOWS IN STORM DRAINAGE NETWORKS[1]

by

D. L. Fread[2]

Abstract. Unsteady flow routing models used for real-time flow predictions or storm drainage network design must be capable of simulating mixed flows, i.e., flows which change with time and location along the storm conduit from free surface unsteady flow to pressurized flow and conversely. A general description of such a mathematical model is presented. The model will also properly predict backwater effects, flow reversals, and surface flooding effects. The model is based on the complete one-dimensional equations of unsteady flow which are solved by a weighted four-point implicit finite difference scheme. At each time step, a system of nonlinear algebraic difference equations are solved for the unknown flow and water surface elevation at specified locations along the storm drainage network using Newton-Raphson iteration and a specially constructed Gaussian elimination matrix technique which has efficient computational properties. The model utilizes a very narrow fictitious slot emanating from the top of the storm conduit to effect the proper wave propagation speed when the conduit becomes pressurized. The conduit may be circular or have an arbitrary shape. The storm drainage network may be a single conduit or a dendritic network of conduits including multiple outlets and bypasses. Complex internal hydraulics due to the presence of manholes, overflow weirs, off-line detention storage basins and pumping stations are simulated via appropriate equations introduced within the system of flow equations as internal boundary conditions.

Introduction

When a storm drainage network receives flows from a large storm event, the flow regime may change from an initial free surface flow everywhere within the system to a mixed flow regime in which portions of the system experience pressurized flow as the conduit continues to fill. Unsteady flow routing models used for real-time flow prediction or storm drainage network design must be capable of simulating such mixed flows, i.e., flows which change with time and location along the storm drains from free surface unsteady flow to pressurized flow and conversely. During the time when the flow is in the free surface regime, many large trunk storm conduits are subject to backwater and reverse flow effects due to the very mild slopes dictated by outlet conditions and topography. An unsteady flow routing model must be able to properly simulate such effects which significantly deplete available storage within the system and help to actuate the pressurization of the system. For this reason, the routing model must be of either the dynamic wave or diffusion type. The degree of

[1]1984 Intl. Symp. on Urban Hydrology, Hydraulics, and Sediment Control, Univ. of Kentucky, Lexington, KY, July 23-26, 1984, pp. 215-222.

[2]Sr. Res. Hydro., Hydro. Res. Lab., Natl. Weather Svc., NOAA, Silver Spring, MD 20910.

of unsteadiness of the flow within a storm drainage network due to rapidly increasing inflows and pressurization waves favors the use of a dynamic wave type routing model.

The purpose of this paper is to present a description of an unsteady flow routing model capable of simulating mixed flows, backwater effects, reverse flows, and rapidly varying transient flows in mild sloping storm drainage networks. The model is of the dynamic wave type and is based on the complete one-dimensional equations of unsteady flow (Saint-Venant equations) which are solved by a weighted four-point implicit finite difference scheme. At each time step, a system of nonlinear algebraic difference equations are solved for the unknown flow and water surface elevation at specified locations (nodes) along the storm water conduit by using Newton-Raphson iteration coupled with a specially constructed Gaussian elimination matrix technique which provides very efficient computational properties. The storm conduit may be of circular or arbitrary shape. The network may consist of a single pipe or a dendritic system of pipes including multiple outlets, bypasses, and cross-connections. The model's mixed flow capability is made possible by introducing a very narrow fictitious chimney or slot in the top of the pipe after than introduced by Cunge and Wegner and referred to as the Preissmann slot.[1] Complex internal hydraulics due to the presence of manholes, overflow weirs, time-dependent gates, off-line detention storage basins and pumping stations, and drop inlet structures are simulated via appropriate equations introduced within the system of nonlinear flow equations as internal boundary conditions. The storage and pumping basins are connected to the storm drainage system via weir overflows where submergence effects and possible flow reversals are considered. Each pump has its individual operating characteristics. Surface flooding, occurring at manholes when the conduit is pressurized, is either considered lost to the system or returns undiminished in volume after flooding a prescribed area. The downstream outlet(s) of the storm drainage system are governed by a specified head-discharge rating or water elevation-time history.

Relation to Previous Work

A variety of flow routing models have been developed specifically for storm drainage systems. Differences in the models can be attributed to their degree of simplicity, required computational effort, computational techniques utilized, and their general applicability.

Some very simple models include the Chicago Hydrograph Method reported by Tholin and Keefer, the TRRL model by Watkins, and the ILLUDAS model described by Terstriep and Stall.[2,3,4] These routing models are based on steady flow hydraulics. Other simple models include the SURKNET model, presented by Pansaic and Yen, and the Wallingsford model described by Price.[5,6] Both of these models handle mixed flow. SURKNET uses a kinematic routing technique for free surface flow and treats the pressurized flow with a cascade solution technique from upstream to downstream. The Wallingsford model uses the Muskingum-Cunge routing method for free surface flows and solves the pressurized flow equations simultaneously for all pipe segments in which pressurization occurs.[7] Time steps are 15-30 seconds for free surface flow and 1 second for pressurized flow. Both of these models consider surface flooding due to surcharged manholes, but neither consider backwater effects or flow reversals in the free surface regime. Wood and Heitzman have developed an explicit finite element model for the complete one-dimensional equations of pressurized flow in a dendritic network of conduits.[8] They also developed two simpler models for pressurized flow only.

A few models are based on the complete one-dimensional unsteady flow (Saint-Venant) equations for the free surface regime with special treatment of the pressurized flow regime. In the popular SWMM model, the free surface flow routing uses a kinematic technique or an alternative method described by Roesner et al. based on an explicit finite difference solution of the Saint-Venant equations.[9] Pressurized flow is handled through a modified continuity relationship applied at each manhole. Any surface flooding is assumed to be lost from the system. Extremely small time steps are required for numerical stability. Song et al. have developed a model which uses an explicit finite difference solution of the characteristic form of the one-dimensional unsteady flow equations with the small disturbance celerity changed as the conduit undergoes pressurization.[10] A shock fitting technique, similar to that used by Cunge et al., is used at the interface of the two flow regimes.[11] Numerical stability restricts the allowable time step to about 0.5 - 2 seconds, depending on the propagation speed of the pressurization wave. Surface flooding is not considered. Two proprietary models developed in Europe which are not readily available have been reported in the literature. The French model, CAREDAS, developed by Sogréah, and the SYSTEM II SEWER model, developed at the Danish Hydraulics Institute, use the so-called Preissmann slot technique to enable the Saint-Venant equations to handle both free surface and pressurized flow.[12,13] Song et al. also have a model which uses the slot technique along with an explicit method of characteristics solution of the Saint-Venant equations.[10]

The model presented in this paper is similar to the European models in that a fictitious slot is used to enable the Saint-Venant equations to handle pressurized flow. It differs in the way the Saint-Venant equations are expressed in finite difference form and in the way a network of conduits are computationally treated. Other differences include the internal boundary equations used to simulate selected critical flow drop inlets, off-line detention storage and pumping basins, surface flooding, and junction losses. This model is a significant extension of the work of Bhattacharyya and Fread.[14] It is an optional modification to the National Weather Service DWOPER model which has received widespread use within the engineering community for a variety of unsteady free surface flow routing applications.[15] Due to the basic DWOPER model having a modular construction with specialized subroutines and variable array sizes, the modifications required for mixed flow routing in conduit networks has little effect on its required computational resources.

Model Description

Governing Equations

The mixed flow routing model is based on the one-dimensional Saint-Venant equations of unsteady flow, i.e.,

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} - q = 0 \tag{1}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial (Q^2/A)}{\partial x} + gA\left[\frac{\partial h}{\partial x} + S_f\right] = 0 \tag{2}$$

where: $$S_f = \frac{n^2|Q|Q}{2.2\ A^2\ R^{4/3}} \tag{3}$$

and $$n = 0.0735\ f^{1/2}\ d^{1/6} \tag{4}$$

in which Q = flow (ft^3/sec), A = cross-sectional area (ft^2), q = lateral inflow (ft^2/sec), x = distance along the conduit (ft), t = time (sec), g = acceleration due to gravity (ft/sec^2), h = water surface elevation (ft), S_f = friction slope (ft/ft), R = hydraulic radius (ft) = A/P where P is the wetted perimeter (ft) of the cross-section, n = Manning roughness coefficient, d = the conduit diameter (ft), and f = the Darcy friction factor.

Eq. (1) which conserves mass, and Eq. (2) which conserves momentum, are quasi-linear hyperbolic partial differential equations. They are solved herein by using a numerical approximation procedure, the weighted four-point implicit finite difference scheme which obtains solutions of h and Q at discrete points in space (x) and time (t). In this scheme, the following approximations are used:

$$\frac{\partial K}{\partial t} = \left(K_i^{j+1} + K_{i+1}^{j+1} - K_i^j - K_{i+1}^j\right)/2\Delta t_j \tag{5}$$

$$\frac{\partial K}{\partial x} = \theta\left(K_{i+1}^{j+1} - K_i^{j+1}\right)/\Delta x_i + (1-\theta)\left(K_{i+1}^j - K_i^j\right)/\Delta x_i \tag{6}$$

$$K = \theta\left(K_{i+1}^{j+1} + K_i^{j+1}\right)/2 + (1-\theta)\left(K_{i+1}^j + K_i^j\right)/2 \tag{7}$$

in which K represents any variable (Q, h, q, A, S_f) in Eqs. (1-2) and the (i, i+1) subscripts represent locations (nodes) along the conduit in the x-direction and the superscripts (j, j+1) represent successive points in time. The recommended value of the weighting factor (θ) is in the range $0.55 \le \theta \le 0.65$.

Substituting Eqs. (5-7) into Eq. (1) and Eq. (2), respectively, yields the following difference equations after some minor rearranging of terms:

$$\theta\left(Q_{i+1}^{j+1} - Q_i^{j+1} - \bar{q}_i^{j+1}\Delta x_i\right) + (1-\theta)\left(Q_{i+1}^j - Q_i^j - \bar{q}_i^j\Delta x_i\right) + 0.5\ \Delta x_i/\Delta t_j\left(A_i^{j+1} + A_{i+1}^{j+1} - A_i^j - A_{i+1}^j\right) = 0 \tag{8}$$

$$0.5\ \Delta x_i/\Delta t_j \left(Q_i^{j+1} + Q_{i+1}^{j+1} - Q_i^j - Q_{i+1}^j\right) + \theta \Big[(Q^2/A)_{i+1}^{j+1} - (Q^2/A)_i^{j+1} + g\bar{A}_i^{j+1}\left(h_{i+1}^{j+1} - h_i^{j+1} + \bar{S}_{f_i}^{j+1}\,\Delta x_i\right)\Big] + (1-\theta)\Big[(Q^2/A)_{i+1}^j - (Q^2/A)_i^j + g\bar{A}_i^j\left(h_{i+1}^j - h_i^j + \bar{S}_{f_i}^j\,\Delta x_i\right)\Big] = 0 \tag{9}$$

where: $$\bar{A}_i = (A_i + A_{i+1})/2 \tag{10}$$

$$\bar{S}_{f_i} = n_i^2\,|\bar{Q}_i|\,\bar{Q}_i/(2.21\ \bar{A}_i^2\ \bar{R}_i^{4/3}) \tag{11}$$

$$\bar{Q}_i = (Q_i + Q_{i+1})/2 \tag{12}$$

$$\bar{R}_i = \bar{A}_i/\bar{P}_i \tag{13}$$

$$\bar{P}_i = (P_i + P_{i+1})/2 \tag{14}$$

Eqs. (8-9) are nonlinear with respect to the unknowns $\left(Q_i^{j+1}, Q_{i+1}^{j+1}, h_i^{j+1}, h_{i+1}^{j+1}\right)$. The terms A and P are known functions of h, and S_f is a known function of Q and h. All terms with a superscript (j) are known from the initial conditions or a previous solution at the j^{th} time.

Initial Conditions

The initial conditions are the values of h_i^j and Q_i^j for all nodes (i = 1, 2, ..., N) along the x-axis at time (t = 0) or j = 1. In this model, these may be obtained by either specifying them as input to the model or by letting the model compute them on the basis of the assumption of steady, spatially varied flow at t = 0.

When they are specified as input, the model uses them to obtain solutions to Eqs. (8-9), and then uses the solutions to obtain other solutions. This process is repeated a number of times until any small errors in the initial values have been damped out by the successive solutions. If the initial values do not contain large errors, this process converges; however, it may not if the errors are too large.

A preferable method is to let the model compute the initial conditions using the following steady gradually varied flow difference equation, i.e.,

$$(Q^2/A)_{i+1} - (Q^2/A)_i + g\bar{A}_i\,(h_{i+1} - h_i + \bar{S}_{f_i}\,\Delta x_i) = 0 \tag{15}$$

in which $\bar{A}_i$ and $\bar{S}_{f_i}$ are defined by Eqs. (10-11). The computations proceed in the upstream direction from a specified value for h_{i+1} at the most downstream point in the system. Thus, Eq. (15) is recursively solved for h_i; since A_i and $\bar{S}_{f_i}$ are nonlinear functions of h_i, Eq. (15) is nonlinear and is solved by the Newton-Raphson iterative method for a single equation. The model determines all the Q values by a simple summation process using inflow values at $t = 0$ for all specified inflow points. The h and Q values can be determined in this way for either a single conduit or a complex dendritic network of conduits.

Boundary Conditions

Boundary conditions are specified values of either h or Q, or a known relation between them, at all the upstream-most nodes in a network of conduits and at the most downstream node. In this model, the upstream boundary conditions are known inflows as a function of time (specified discharge hydrographs). The downstream boundary can be a known water surface elevation as a function of time such as for submerged outlets in lakes or estuaries. Also, the downstream boundary can be a known relation between Q and h such as normal flow, critical flow, etc. which is input in the form of a table of Q and h values.

Solution Technique

The governing finite difference approximating equations, Eqs. (8-9), can be solved once the initial conditions are obtained and all boundary conditions specified. Eqs. (8-9) cannot be solved in an explicit or direct manner for the four unknowns since there are only two equations. However, if Eqs. (8-9) are recursively applied to each Δx segment along a single conduit or network of conduits, a total of (2N - 4J - 2) equations with 2N unknowns can be formulated where N is the total number of nodes and J is number of junctions. Then, prescribed boundary conditions, one at the upstream end of each conduit and one at the downstream end of the main conduit, and three compatibility equations for each junction in the network, provide the necessary additional equations required for a determinate system. The resulting system of 2N nonlinear equations with 2N unknowns is solved by a functional iterative procedure, the Newton-Raphson method for a system of equations.[16]

Computations for the iterative solution of the nonlinear system are begun by assigning trial values to the 2N unknowns. Substitution of the trial values into the system of nonlinear equations yields a set of 2N residuals. The Newton-Raphson method provides a means for correcting the trial values until the residuals are reduced to a suitable tolerance level. This is usually accomplished in one or two iterations through use of linear extrapolation for the first trial values. A system of 2N x 2N linear equations relates the corrections to the residuals and to a Jocobian coefficient matrix composed of partial derivatives of each equation with respect to each unknown variable in that equation. The coefficient matrix of the linear system had a banded structure which allows the system to be solved by a compact penta-diagonal Gaussian elimination algorithm which is very efficient with respect to computing time and storage. The required storage is 2N x 4 and the required number of computational steps is approximately 38N.[17] In the case of networks, the required number of computational steps is (102 + 46J)N when using a specially constructed matrix technique which minimizes the number of off-diagonal elements due to the junctions and operates only on the non-zero elements in the Jacobian matrix.[18]

Conduit Geometry

The conduit may be circular or arbitrary shape. A very narrow fictitious slot or chimney emanates from the top of the conduit.

Circular Conduit Properties. Circular conduits have the following geometric properties which are used in the model:

$$\omega = \pi + 2 \sin^{-1} [(y-r)/r] \;\ldots\ldots\; y < d \tag{16}$$

$$P = r\,\omega \tag{17}$$

$$A = 0.125\, d^2 (\omega - \sin \omega) \;\ldots\ldots\; y < d \tag{18}$$

$$A = 0.25\, \pi\, d^2 \;\ldots\ldots\; y \geq d \tag{19}$$

$$B = d \sin (0.5\, \omega) \;\ldots\ldots\; y < d \tag{20}$$

$$B = b \;\ldots\ldots\; y \geq d \tag{21}$$

$$dP/dy = 2/\sqrt{1 - [(y-r)/r]^2} \;\ldots\ldots\; y < d \tag{22}$$

$$dP/dy = 0 \;\ldots\ldots\; y \geq d \tag{23}$$

in which ω = the central angle (rad) subtended by two lines drawn from the center of the conduit to the two points where the water surface intersects the conduit perimeter, y = the depth of flow in the conduit (ft), r = the radius of the circular conduit (ft), P = the wetted perimeter (ft), A = the wetted cross-sectional area (ft^2), B = top width of the wetted area (ft), d = the diameter (ft) of the conduit, and b = the chimney width.

Arbitrary-Shaped Conduits. Conduits with an arbitrary shape whose geometric properties may not be readily defined analytically as with circular shapes may be modeled by inputting a table of the conduit width as a linear function of the depth, i.e., (B_k, Y_k) where $k = 1, 2, \ldots N_s$; N_s is a sufficient number of widths to fully describe the geometry including the fictitious slot. The wetted perimeter and area corresponding to each are calculated initially (one time only) as follows:

$$P_k = P_{k-1} + 2\sqrt{(\Delta B_k)^2 + (\Delta Y_k)^2} \tag{24}$$

where: $$\Delta B_k = (B_k - B_{k-1})/2 \tag{25}$$

$$\Delta Y_k = Y_k - Y_{k-1} \tag{26}$$

and, $$A_k = \sum_{\ell=1}^{k} 0.5(B_\ell + B_{\ell-1})(Y_\ell - Y_{\ell-1}) \tag{27}$$

In Eqs. (24-27), the k index goes from 2 through N_s; when $k = 1$, $P_1 = B_1$ and $A_1 = 0$. The values of B, P, and A, associated with any depth (y) during the unsteady flow solution, can be computed according to the following linear relations:

$$B = B_k + (B_{k+1} - B_k)(y - Y_k)/(Y_{k+1} - Y_k) \tag{28}$$

$$P = P_k + (P_{k+1} - P_k)(y - Y_k)/(Y_{k+1} - Y_k) \tag{29}$$

$$A = A_k + 0.5(B + B_k)(y - Y_k) \tag{30}$$

Eqs. (28-30) are applicable when $Y_k \leq y \leq Y_{k+1}$.

Fictitious Slot. From the top of the conduit, a hypothetical chimney or slot extends upward.[1,11] The purpose of the slot is to provide a cross-sectional area whose top width (b) is very small for all flow depths greater than the top of the conduit. The small top width is required to produce the proper celerity for pressurized flow as computed from the gravity wave celerity equation, i.e.,

$$c = \sqrt{gA/B} \tag{31}$$

in which c is the celerity (ft/sec) of a gravity wave. For a circular conduit, Eq. (31) can be used to compute the required chimney width by replacing B with b and c with a (the celerity of a pressure wave). Solving Eq. (31) for b yields the following relation:

$$b = 0.25\, g\, \pi \left[\frac{d}{a}\right]^2 \tag{32}$$

The pressure wave celerity (a) can be computed from the properties of the conduit and the storm water which may contain some entrained air.[19] Thus,

$$a = \left[\frac{K/\rho}{1 - Kdc_1/(Ee)}\right]^{1/2} \tag{33}$$

where: $$K = \frac{K_w}{1 + V_a\left[\frac{K_w}{K_a} - 1\right]} \tag{34}$$

$$\rho = \rho_a V_a + \rho_w V_w \tag{35}$$

in which K = the bulk modulus of elasticity of the flowing water (lb/ft^2), ρ = the bulk density of the flowing water (lb sec^2/ft^4), d = the conduit diameter (ft), E = Young's modulus of elasticity for the conduit (lb/ft^2), e = the conduit wall thickness (ft), K_w = modulus of elasticity for water (lb/ft^2), K_a = modulus of elasticity for air (lb/ft^2), V_a = the ratio of air volume to the total volume, V_w = the ratio of water volume to the total volume, ρ_a = density of air (lb sec^2/ft^4), and ρ_w = density of water (lb sec^2/ft^4). In Eq. (33), the term c_1 is given by the following:

$$c_1 = \alpha_o + \alpha_1 \beta \tag{36}$$

in which for thick-walled conduits (e/d $\geq$ 25) $\alpha_o = \mu$ and $\alpha_1 = 1$, while for thin-walled conduits (e/d < 25):

$$\alpha_o = 2\ e/d\ (1 + \mu) \tag{37}$$

$$\alpha_1 = d/(d + e) \tag{38}$$

in which μ = the Poisson ratio (lateral stress/axial stress). The β term in Eq. (36) depends on the rigidity of the conduit with respect to axial expansion, i.e, $\beta = 1.0$ if expansion joints are used throughout the length of the conduit, $\beta = 1 - \mu^2$ if it is anchored everywhere, and $\beta = 1.25 - \mu$ if it is anchored only at the upstream end.

Entrained air greatly affects the pressure wave celerity; as the air content increases from 0.0 to 1.0 percent, the celerity can decrease from about 4000 ft/sec to 700 ft/sec. Of course, field measurements of pressure celerity would be preferable for determining the chimney width (b) from Eq. (32).

Internal Boundaries

Locations along the conduit network where the Saint-Venant equations are not applicable are called internal boundaries. Such locations include manholes, junctions, drop inlets where critical flow occurs, connections to off-line detention storage basins, and/or pumping stations. The internal boundary consists of two equations which replace the two Saint-Venant equations. These equations relate Q and h at the entrance(s) to a short Δx reach within which the two internal boundary equations describe the hydraulics.

Manholes. Manholes are located where the conduit changes size, slope, and/or direction or where there is a junction of two or three conduits. The two internal boundary equations are:

$$Q_i^{j+1} + \bar{m}\, Q_{i'}^{j+1} - Q_{i'+1}^{j+1} + Q_{in} - Q_w - \Delta s/\Delta t = 0 \tag{39}$$

$$h_i^{j+1} - h_{i'+1}^{j+1} - h_{\ell_i} = 0 \tag{40}$$

$$h_{i'}^{j+1} - h_{i'+1}^{j+1} - h_{\ell_{i'}} = 0 \tag{41}$$

in which Q_i = the inflow (ft^3/sec) to the manhole from the upstream conduit, $Q_{i'}$ = the inflow from the branch conduit (m = 0 if there is no branch conduit, otherwise m = 1), $Q_{i'+1}$ = the outflow from the manhole through the exiting downstream conduit (when there is no branch conduit, $i' = i$), h_i, $h_{i'}$, and $h_{i'+1}$ are the water surface elevations (ft) of the upstream, branch, and downstream conduits, respectively, Q_{in} = the surface inflow to the manhole which is a specified function of time, Q_w = the flow entering or leaving the manhole via a weir-type control, $\Delta s/\Delta t$ = the change of storage associated with the manhole during a Δt time step, h_{ℓ_i} and $h_{\ell_{i'}}$ are the head losses incurred by the incoming and exiting flows.

The weir flow (Q_w) is further defined as follows:

$$Q_w = C\, C_s\, (\bar{h} - h_w)^{3/2} \; \ldots\ldots \; \bar{h} > h_w \tag{42}$$

where: $$\bar{h} = \left(h_i^{j+1} + m\, h_{i'}^{j+1} + h_{i'+1}^{j+1}\right) / (m + 2) \tag{43}$$

$$C = 3.2\, L_w \tag{44}$$

$$C_s = \left(1 - h_r^2\right)^{1/2} \tag{45}$$

$$h_r = (\bar{h} - h_w) / (h_t - h_w) \; \ldots. \; \bar{h} \text{ and } h_t > h_w \tag{46}$$

$$h_r = 0 \; \ldots. \; \bar{h} \text{ or } h_t < h_w \tag{47}$$

in which h_w is the crest elevation of the weir, $\bar{h}$ is the water elevation within the manhole as given by Eq. (43) in which m = 1 when a branching conduit exists and m = 0 where there is no branch conduit, C is the weir discharge coefficient, L_w is the length (ft) of the weir crest, C_s is the submergence correction factor given by Eq. (45), h_r is given by either Eq. (46) or Eq. (47) in which h_t is the water elevation downstream of the weir.

The change in manhole storage is given by the following:

$$\Delta s = 0.25\, \pi\, d_m^2\, (\bar{h} - \bar{h}') \; \ldots. \; \bar{h} \le h_m \tag{48}$$

in which d_m = the diameter (ft) of the manhole, $\bar{h}'$ = the water elevation within the manhole at time (t - Δt), and h_m = the elevation of the top of the manhole.

The junction losses h_{ℓ_i} and $h_{\ell_{i'}}$ in Eqs. (41-42) are approximated as follows:

$$h_{\ell_i} = K \left[\frac{V_i^j + V_{i'+1}^j}{2} \right]^2 / 2g \tag{49}$$

$$h_{\ell_{i'}} = K \left[\frac{V_{i'}^j + V_{i'+1}^j}{2} \right]^2 / 2g \tag{50}$$

in which V_i^j, $V_{i'}^j$, and V_{i+1}^j are the velocities at time $(t - \Delta t)$ of the upstream, branch, and downstream conduits, respectively. The head loss coefficient (K) varies from about 0.10 to 0.30 for straight-flow-through junctions depending on junction geometry.[20] The head losses are assumed to be negligible or compensated by small drops in the inlet-outlet inverts within the manhole for all flows less than about three-fourth of the outlet conduit's full discharge capacity.

Eq. (50) is not used when the branch conduit does not exist; in this case, the subscript (i') is the same as i in Eq. (49).

Surface Flooding. Flooding of the surrounding surface area above the top of a manhole is treated by either of two methods for each manhole. If the first, it is assumed that the flooding occurs as the water elevation of the manhole increases above the top of the manhole (h_m). When this occurs, Eq. (48) becomes:

$$\Delta s = d_s^2 (\bar{h} - \bar{h}') \;\ldots.\; \bar{h} > h_m \tag{51}$$

in which d_s is the length (ft) of the flooded surface area which is assumed to be represented by a square. It is further assumed that the surface flooding does not result in any permanent loss of volume to the conduit system; thus, all of the flood waters return to the system as the water level in the manhole diminishes. In the second method, the surface flooding occurs when the manhole water elevation $(\bar{h})$ becomes greater than h_m, and the overtopping waters are assumed to be permanently lost to the conduit system. The exiting flow is denoted by Q_w in Eq. (39); it is computed from Eq. (42) rewritten as follows:

$$Q_w = 3.1\, \pi\, d_m (\bar{h} - h_m)^{3/2} \;\ldots.\; \bar{h} > h_m \tag{52}$$

Submergence corrections are neglected in Eq. (52).

Off-Line Detention Storage and Pumping Basins. Detention basins which temporarily store excess waters are connected to the conduit system via a manhole junction. The stored waters are removed from the system via the term Q_w in Eq. (39) and computed using Eqs. (42-47). Q_w may be plus or minus according to the relative values of $\bar{h}$ and h_t. The term h_t used in

Eq. (46) is the water surface elevation of the detention basin; it is computed from a storage balance of the detention basin at time (t - Δt).

$$S^{j} = S^{j-1} + \left(Q_{w}^{j} - Q_{p}^{j}\right) \Delta t_{j} \tag{53}$$

in which S^j and S^{j-1} are the surface storage area (ft²) of the detention basin at times (t - Δt) and (t - 2Δt), respectively. Q_p represents outflow from the basin due to pumping.

If the basin has one or more pumps, Q_p represents the total discharge from the pumps as determined from a specified tabular head-discharge rating for each pump. Also, each pump has a specified elevation of the detention basin water surface at which the pump starts and one at which it stops pumping. The head (H) used to enter the head-discharge rating table to determine the pump discharge is:

$$H = h_{p} - h_{t} \tag{54}$$

in which h_p is the average head including friction losses against which each pump discharges, and h_t is the water surface elevation of the detention (pump) basin; h_t is obtained through interpolation of the surface area (S)-elevation (H_t) table which is specified for each detention or pump basin.

Drop Inlets. The invert of a conduit inlet to a manhole may be considerably higher than the other inlet and outlet. Such drop inlets must be treated as an internal boundary since the flow dropping into the manhole is governed by the critical flow equation rather than the Saint-Venant equations. Thus, a very small Δx reach is specified immediately upstream of the conduit outlet where critical flow occurs. The following two internal boundary equations are used:

$$Q_{i}^{j+1} - Q_{i+1}^{j+1} = 0 \tag{55}$$

$$Q_{i}^{j+1} = \left(\sqrt{gA^{3}/B}\right)_{i}^{j+1} = 0 \tag{56}$$

If the flow in the vicinity of the drop inlet starts to be pressurized such that the critical flow point becomes submerged, Eqs. (55-56) are replaced by the Saint-Venant equations. The necessity of such a change is continually checked during the computations by comparing the manhole water elevation $(\bar{h})$ against the water elevation $\left(h_{i}^{j+1}\right)$.

Computational Sensitivity

The model was tested for its sensitivity to the distance step (Δx), the time step (Δt), and the chimney width (b). The computations were found to be insensitive to a range of Δx values representing the usual spacing of manholes. The time step recommended for free surface flow using the 4-pt. implicit solution of the Saint-Venant equations applied to a circular conduit is the following:

$$\Delta t \leq T_p/M \tag{57}$$

$$M = 2.67\left[1 + 3.78\ n/(d^{1/6}\ S^{1/2}\right] \tag{58}$$

in which T_p is the duration of the rising limb of the inflow hydrograph, n is the Manning n, d is the storm conduit diameter, and S is the bottom slope of the storm conduit. The units of Δt and T_p are the same (hr or sec).

Sensitivity of the computed results to variations in the time step (Δt) was tested on a number of storm drainage networks. One such system is shown in Fig. 1. It consists of six manholes, each with an inflow hydrograph having a time of rise of 0.25 hr.

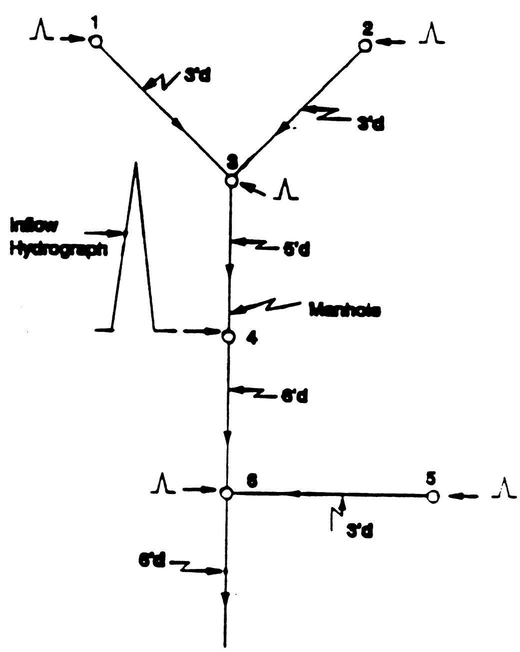

Figure 1. Test Storm Drainage System

The system was tested with and without surcharging; the latter condition was developed by increasing the magnitude of the peak of the inflow hydrograph at manhole no. 4. Conduit diameters vary from 3 to 6 ft, and the invert slope of each is 0.00075 ft/ft. The distance (Δx) between each manhole is 580 ft. The Δt computed from Eq. (57) is 110 sec for $\theta = 0.60$, $\varepsilon = 0.97$, $n = 0.010$, and $T_p = 0.25$. The computed maximum water surface elevations, discharges, and times of occurrence at manhole nos. 1, 4, 5, and 6 were compared with those computed using a very small time step of 1.8 sec. The comparative variations expressed as percent are shown in Table 1. The errors are quite small (less than about 5 percent) for all Δt values less than or equal to 110 sec, that given by Eq. (57). As Δt is considerably increased above this value, the error becomes significant. For larger T_p values, the time step which would produce insignificant errors could be increased.

Table 1. Average Errors (Percent) Associated with the Maximum Computed Values at Manhole Nos. 1, 4, 5, and 6 for Various Time Steps

Δt (sec)	Time of Maximums	Water Surface Elevation (Head)	Discharge
1.8	0.	0.	0.
18.	0.7	1.0	1.0
36.	0.8	1.2	1.7
110.	6.0	4.0	5.0
180.	11.9	6.5	8.4
450	6.7	29.8	28.1

The computational requirements on an IBM 360/195 are approximately 0.009 sec per manhole per time step and on a PR1ME 750 computer this is increased by a factor of about seven.

Summary and Conclusions

A nonlinear weighted implicit finite difference model based on the complete Saint-Venant equations of unsteady flow has been developed. The model is capable of simulating free surface and/or pressurized flows, backwater effects, and flow reversals in a single storm conduit or a complex dendritic network of conduits having cross-connections and multiple outlets. The storm conduit may have a circular or an arbitrary shape. Manhole effects (junctions of two or three conduits, head losses, storage, surface flooding with or without loss of volume), detention basin storage, pumping stations with one or more pumps with individual operating characteristics, and critical flow drop-inlets are each simulated by appropriate equations introduced as internal boundaries and solved simultaneously with the Saint-Venant finite difference equations and external boundary equations.

Pressurized flow is conveniently treated with the Saint-Venant equations by utilizing a very narrow hypothetical chimney (slot) which extends upward from the top of the storm conduit. This feature enables the wave celerity to change from a gravity wave to a pressure wave as the flow depth increases and reaches the top of the conduit whereupon the computed depth represents the pressure head. Conversely, the flow regime at any location or at any time may change from pressurized to free surface. An equation is presented for computing the proper chimney width which depends on the properties of the storm conduit and the storm water.

Numerical testing of the model indicates a desirable insensitivity to reasonable ranges of the distance step (Δx) and the time step (Δt). The former can be the normal distance between manholes where flow and conduit properties may change, and the latter is selected to be less than or equal to that given by an equation which relates the storm drains' hydraulic properties, the rate of inflow to the system, and the acceptable numerical solution error.

References

1. J. A. Cunge and M. Wegner, "Numerical Integration of Barré de Saint-Venant's Flow Equations by Means of an Implicit Scheme of Finite Differences. Applications in the Case of Alternately Free and Pressurized Flow in a Tunnel," La Houille Blanche, No. 1, pp. 33-39, 1964.

2. A. L. Tholin and C. J. Keefer, "The Hydrology of Urban Runoff," Transactions, ASCE, Vol. 125, pp. 1308-1379, 1960.

3. L. H. Watkins, "A Guide for Engineers to the Design of Storm Sewer Systems," Road Research Technical Paper No. 55, Dept. of Science of Industry Research, Great Britain, 1963.

4. M. L. Terstriep and J. B. Stall, "The Illinois Urban Drainage Area Simulator, ILLUDAS," Bulletin 58, Illinois State Water Survey, Urbana, Illinois, 1974.

5. N. Pansaic and B. C. Yen, "Kinematic-Wave Modelling of Storm Sewer Systems with Surcharge," Proceedings of the Second International Conference on Urban Storm Drainage, Vol. I, Hydraulics and Hydrology, ed. by B. C. Yen, University of Illinois, Urbana, Illinois, pp. 193-202, June 1981.

6. R. K. Price, "A Simulation Model for Storm Sewers," Second International Conference on Urban Storm Drainage, Urbana, Illinois, pp. 184-192, June 1981.

7. J. A. Cunge, "On the Subject of a Flood Propagation Method," Journal of Hydraulics Research, IAHR, Vol. 7, pp. 205-230, 1969.

8. D. J. Wood and G. C. Heitzman, "Hydraulic Analysis of Surcharged Storm Sewer System," Research Report No. 137, University of Kentucky Water Resources Research Institute, Lexington, Kentucky, 143 pp., 1983.

9. L. A. Roesner, R. P. Shubinski, and J. A. Aldrich, "Storm Water Management Model User's Manual Version III, Addendum I Extran," U.S. Environmental Protection Agency, Cincinnati, Ohio, 1981.

10. C. C. S. Song, J. A. Cardle, and K. S. Leung, "Transient Mixed-Flow Models for Storm Sewers," Journal of Hydraulic Engineering, ASCE, Vol. 1109, No. 11, pp. 1487-1503, November 1983.

11. J. A. Cunge, F. M. Holly, Jr., and A. Verweg, Practical Aspects of Computational River Hydraulics, Pitman Advanced Publishing Program, Boston, Massachusetts, 1980.

12. Sogréah, "Mathematical Model of Flow Simulation in Urban Sewerage Systems, CAREDAS Program," Societeté Grenobloise d'Etudes et d'Applications Hydrauliques, Grenoble, 1973.

13. N. E. Hoff-Clausen and A. Kej., "System II Sewer -- A Storm Sewer Model," Proceedings of the Second International Conference on Urban Storm Drainage, Vol. I, Hydraulics and Hydrology, ed. by B. C. Yen, University of Illinois, Urbana, Illinois, pp. 137-145, June 1981.

14. A. K. Bhattacharyya and D. L. Fread, "Storm Surge Prediction Model and its Application to a Combined Trunk Sewer System," Proceedings of Storm Water and Water Quality Models Users Group Meeting, ed. by P. Beron and T. Barnwell, Ecole Polytechnique De Montreal, Montreal, Quebec, pp. 32-46, November 1983.

15. D. L. Fread, NWS Operational Dynamic Wave Model," Proceedings of 26th Annual ASCE Hydraulics Division Specialty Conference, University of Maryland, College Park, Maryland, pp. 455-464, August 1978.

16. M. Amein and C. S. Fang, "Implicit Flood Routing in Natural Channels," Journal of Hydraulics Division, ASCE, Vol. 96, No. HY12, pp. 2481-2500, December 1970.

17. D. L. Fread, "Discussion of Implicit Flood Routing in Natural Channels by M. Amein and C. S. Fang," Journal of Hydraulics Division, ASCE, Vol. 97, No. HY7, pp. 1156-1159, July 1971.

18. D. L. Fread, "Computational Extension to Implicit Routing Models," Proceedings of Conference on Frontiers in Hydraulic Engineering, ed. by H. T. Shen, Massachusetts Institute of Technology, Cambridge, Massachusetts, pp. 343-348, August 1983.

19. V. L. Streeter and E. B. Wylie, Hydraulic Transients, McGraw-Hill Book Co., New York, 1967.

20. J. Marsalek, "Head Losses at Sewer Junction Manholes," Journal of Hydraualics Division, ASCE, Vol. 110, No. HY8, pp. 1150-1154, August 1984.